

Proposal to Improve the Design of a Brake Drum Cooling System for a 650 Hp Drawworks Through Thermal Analysis and Computational Fluid Dynamics Simulation

E. Fierro¹, H. Falcones², L. Ramirez^{3*}

Abstract — The following work presents a proposal to improve the brake drum cooling system of a 650 HP drawworks, which is crucial in oil well workover operations. To achieve this, a computational fluid dynamic (CFD) and thermal analysis was performed using ANSYS Student software, where the current system was modeled and different cooling scenarios were simulated. The behavior of the cooling fluid was evaluated, and a more efficient solution including a heat exchanger was designed. The results indicated that, with an initial drum temperature of 300 °C, the current water cooling system heats the fluid to 56 °C, after which it goes directly to a 30 m³ tank that serves as a thermal buffer, which, due to its large capacity, manages to cool the water to a range between 31 and 32 °C in a transient state. The proposed improvement, by implementing a heat exchanger, directly reduces the cooling water temperature to 32 °C in a transient state. In addition, the tank volume was improved from 30 m³ to only 6 m³, which considerably reduces water consumption without affecting thermal performance. Finally, it was verified that the centrifugal pump used is adequate to maintain the required pressure throughout the circuit. The following proposal not only improves the efficiency of the system, but also reduces the risk of mechanical failures, saves resources, and improves operative efficiency.

Keywords: Drawworks; drum brake; cooling system; CFD; Ansys; heat exchanger; thermal analysis.

Resumen — El siguiente trabajo presenta una propuesta para mejorar el sistema de enfriamiento de los tambores de freno de un malacate de 650 HP, que es crucial en las operaciones de acondicionamiento de pozos petroleros. Para lograrlo, se ejecutó

un análisis térmico y de dinámica de fluidos computacional (CFD) utilizando el software ANSYS Student, donde se modeló el sistema actual y se simularon diferentes escenarios de enfriamiento. Se evaluó el comportamiento del fluido refrigerante y se diseñó una solución más eficiente que incluye un intercambiador de calor. Los resultados indicaron que, con una temperatura inicial del tambor a 300 °C, el sistema de enfriamiento con agua actual calienta el fluido a 56 °C; luego de esto, va directo a un tanque de 30 m³ que hace las veces de amortiguador térmico, el cual, por su gran capacidad, logra enfriar el agua a un rango entre 31 y 32 °C en estado transitorio. Mientras la mejora propuesta, al implementar un intercambiador de calor, reduce directamente la temperatura del agua de refrigeración a 32 °C en estado transitorio. Además, se mejoró el volumen del tanque, pasando de 30 m³ a solo 6 m³, lo que disminuye considerablemente el consumo de agua sin afectar el rendimiento térmico. Finalmente, se verificó que la bomba centrífuga utilizada es adecuada para mantener la presión necesaria en todo el circuito. La siguiente propuesta no solo mejora la eficiencia del sistema, sino que también reduce el riesgo de fallas mecánicas, ahorra recursos y la eficiencia operativa.

Palabras Clave: Malacate; freno de tambor; sistema de enfriamiento; CFD; Ansys; intercambiador de calor; análisis térmico.

I. INTRODUCTION

THE drawworks is one of the main pieces of equipment used in oil well drilling and workover operations. It is responsible for transmitting power from an internal combustion engine or electric motor to the drill string, which, with the help of a set of pulleys, lifts and lowers the entire pipe string, along with the necessary tools, into an oil well, as shown in Fig. 1. In the case of a 650 HP workover rig, wells with a bottomhole tension of up to 330,000 lbs can be maintained, and when lowering this weight at speeds of approximately 90 ft/min, an efficient braking system is necessary to prevent all the tools from falling and being lost at the bottom of the well. [1]

In heavy-duty industrial lifting systems, such as 650 HP drawworks, braking is a pillar of safety and operability. The brake drums dissipate enormous amounts of thermal energy during loading and unloading maneuvers. If the cooling system is not effective, the drums can overheat, lose braking power, and wear out quickly. Any failure of the cooling system not

This work was supported by the Universidad Politécnica Salesiana. Quito, Ecuador, as part of its priority lines of research in energy efficiency and technologies applied to the industrial sector.

* Corresponding author: lraniirezg@ups.edu.ec.

1. E. Fierro is a Candidate for a Master's Degree in Mechanical Engineering with a major in Mechanical, Hydraulic and thermal Systems Design from the Universidad Politécnica Salesiana - Quito, Ecuador. ORCID number: <https://orcid.org/0009-0003-9021-0617>.
2. H. Falcones is a Candidate for a Master's Degree in Mechanical Engineering with a major in Mechanical, Hydraulic and thermal Systems Design from the Universidad Politécnica Salesiana - Quito, Ecuador. ORCID number <https://orcid.org/0000-0003-3256-0101>.
3. L. Ramirez is with the Mechanical Engineering Department, Universidad Politécnica Salesiana - Quito, Ecuador. ORCID number <https://orcid.org/0000-0003-2569-2974>.

DOI: <https://doi.org/10.29019/enfoqueute.1212>

Associate Editor: Victor Surtaxi

only increases the risk of loss of the operational stability of the machinery but also increases the chances of mechanical failures and unscheduled downtime, dealing a blow to the company's productivity through increased maintenance and repair costs [2].

The drawworks brake system is very important for maintaining safe and efficient operations, which is why there are primary and secondary brake mechanisms, which generally use friction mechanisms between a drum and friction discs to perform the braking process, as shown in Fig. 2 This friction process generates large amounts of heat, which must be adequately dissipated to prevent the drums and discs from overheating, thereby reducing braking efficiency and increasing the risk of an accident.

Study [3] take different 3D brake disc designs and use ANSYS CFX to analyze the various phenomena that occur during braking, such as temperature increase, wear, and noise. The results showed that the minimum temperature of the model was 85,54 °C, while the maximum temperature was 110,46 °C.

In [4] a study of heat distribution was conducted between the disc and the pad during the braking stage of a drilling drawworks. During this study, they performed a simulation in the "Heat Transfer in Solids (ht)" module of COMSOL Multiphysics 5.5 software, considering that heat is generated by the friction between the asbestos brake pads and the 35HNL steel brake discs. The results obtained show that the maximum surface temperature reaches values between 1270 and 1550 K and is reached after 2 to 4 seconds of the braking stage according to the given parameters. After the braking process, the temperature is reduced to 500 - 600 K after approximately 15 seconds.

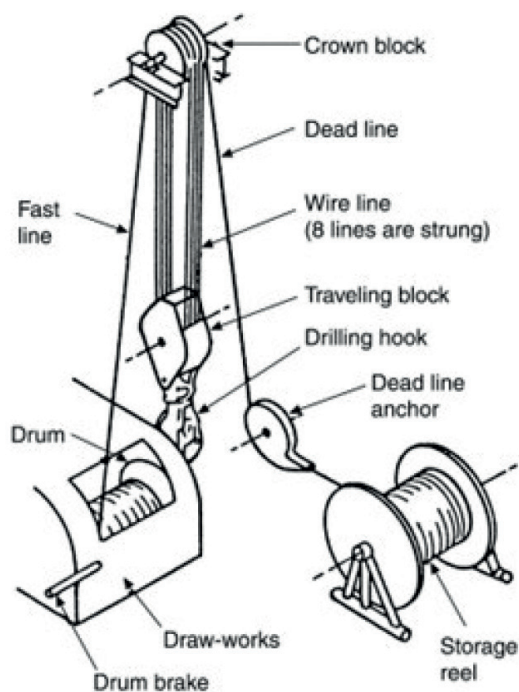


Fig. 1. Hoisting system of a drilling and workover rig [5]

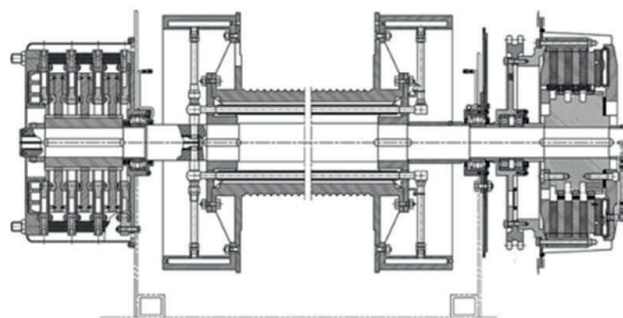


Fig. 2. Drum braking system of drawworks - disc band [5]

Study [6] conducted various experiments to analyze the dynamic friction coefficient and wear between different types of brake pad and disc materials. For this purpose, a base of materials was characterized by whose chemical composition withstands temperatures in the range of 100 to 450 °C with variations of 50 °C, and a wear variation rate between 6.0 and 10.0. The results obtained showed that for materials of types A, B, C, and D, greater wear was observed during the second thermal regime, while for materials E and D, the same wear was observed in both thermal regimes.

Study [7] geometrically documented and analyzed the structure of the first Spanish capstan using 3D modeling and finite element simulation, finding higher stresses in the drum shaft and in the structure that supports them, with simulated stresses equal to 30 MPa and a displacement of 3 mm in the drum and its support. In comparison with the wood, which shows greater displacements equal to 7 mm, stress concentrations were observed in the contact teeth, and the linear elastic behavior was sufficient to validate the design.

Study [8] performed a finite element analysis (FEM) on a thermal fatigue failure of a water-cooled brake disc on an offshore drilling drawworks. For this analysis, they used ABAQUS software, which allowed them to recreate the braking process, and obtain the gradient of the external and internal temperature distribution of the disc. They concluded that the repeated braking process lasting between 3 and 123 seconds, generates thermal stress due to the temperature variation between 186,5 and 295,0 °C, and stresses on the disc varying between 30 and 240 MPa, which is closely related to the appearance of the cracks that caused the failures.

In order to improve performance and reduce temperature rise in the magnetorheological brake during operating conditions. (MR), they designed a water cooling mechanism for the MR multi-disc brake. To achieve this, they designed a cooling mechanism and simulated its operation using COMSOL software. The experimental results show that braking efficiency can reach 96,24 Nm. Additionally, after 20 seconds of continuous braking, the temperature of the brake with the cooling mechanism was approximately 19,78 % lower, confirming that this design has a positive impact [9].

Study [10] developed a coupled thermo-mechanical model and CFD simulations to analyze the thermal behavior of the drum under successive braking, introducing internal channels through which pressurized water circulates. The results showed

a reduction in maximum temperature from 327 °C to 185 °C, a 43 % decrease in thermal fatigue, and a 35 % improvement in friction coefficient stability. The objective of the research was to mitigate overheating, which reduces braking efficiency and causes deformation. Likewise, the numerical model made it possible to determine that the flow rate and geometry of the channels directly influence the efficiency of heat exchange, ensuring uniform convection and prolonging the useful life of the braking system.

This research analyzes the application of a controlled water-cooling system to optimize the thermal performance of eddy current brakes. A coupled thermo-electromagnetic model was developed, and CFD simulations were used to evaluate the heat dissipation generated in high - frequency braking cycles, adjusting variables such as fluid flow rate and temperature. The results showed a reduction in the maximum disc temperature from 350 °C to 190 °C, a 40 % improvement in heat transfer efficiency, and a 30 % decrease in structural deformation. The objective was to prevent overheating, which reduces braking force and accelerates material degradation. The model demonstrated that active water flow control improves forced convection and maintains uniform thermal distribution, ensuring operational stability and longer system life under severe operating conditions. [11].

A study was conducted on the redesign of an industrial winch in a factory in Cuba with the aim of improving the production process by reducing failures and vibrations [12]. They found transmission wear, accelerated corrosion, and fatigue of the steel cable due to crushing. As a result, the chain was replaced with a speed reducer coupled to the motor, the drum was redesigned for cable guidance, and structural analysis was performed using SOLID WORKS 2014, resulting in an efficiency of 0,87 with maximum stresses in the drum of 9,46 MPa, in the chassis 75,6 MPa, and a maximum displacement of 0,0085 mm, eliminating vibrations and noise. The stresses do not exceed the maximum elastic limit of the material (ASTM A36).

Study [13] conducted research on improving heat transfer in minitube heat exchangers using numerical simulations. In particular, the study used the finite volume method to analyze a concentric heat exchanger with internal and external V- shaped corrugations. The researchers simulated countercurrent configurations with hot and cold water as working fluids, setting the inlet temperature at 40 - 15 °C, respectively, and evaluated different mass flow rates between 0,04 and 0,2 kg/s. The results showed that concave corrugations with a phase - change angle of 180° offered a significant improvement in heat transfer rate, demonstrating the effectiveness of these geometries in optimizing thermal efficiency in compact systems.

Study [14] designed an optimized network of heat exchangers for an industrial soybean oil extraction plant, using Pinch Test as a key tool to improve energy efficiency in the solvent separation stage. This methodology enabled a significant reduction in steam 86 % and water 29 % consumption compared to the previous system. Although the implementation of the new equipment increased fixed costs by 31 % due to the greater space required, this increase is economically viable thanks to the rapid return on investment characteristic of industrial processes. The results demonstrate that Pinch Test is a highly effective tool for optimizing energy use in plants with many

years of operation, offering sustainable benefits and improving system efficiency.

Study [15] conducted research in which they analyzed, using numerical simulation, a shell-and-tube heat exchanger using water with 5 % TiO₂ nanoparticles as the hot fluid and water as the cold fluid. The design included 13 copper tubes and five baffles in a steel shell. The results showed that increasing the hot flow rate increased the outlet temperatures by 2,5 to 10 %, while the velocity and pressure fields remained uniform, indicating an efficient design. The addition of nanoparticles improved heat flux by 17,6 % and the convection coefficient by 1,2%, although it reduced the overall heat transfer coefficient by 37,8%.

In [16] studies and simulations were conducted on radiators based on cross-flow tubular heat exchangers, designed for cooling processors, graphics cards, and other hardware components in computer systems. Using ANSYS software, they analyzed various conditions, with coolant inlet temperatures ranging from 75 to 90 °C and variations in mass flow rates. The results indicate that as the inlet temperature of the fluid increases, the outlet temperature also increases, implying greater heat transfer. However, when the mass flow rate was increased, a decrease in heat rejection capacity was observed, highlighting the influence of operating conditions on the thermal efficiency of the cooling system.

A mesh independence study was performed in a CFD simulation of recirculation flow in order to obtain accurate results [17]. STAR-CCM software was used, ANSYS Meshing geometry and mesh design software was used to geometrically design inlet and outlet ducts with a diameter of 0,016 m and a length of 0,25 m, a central box measuring 0,06 m x 0,4 m with a constant velocity of 0,1449 m/s, and a pressure output of 0 Pa. The meshes compared were 1 900,000 cells, with a coupling length of 0,246 m, which was sufficient to obtain good results and a reliable simulation, the coupling point being the key parameter for validating the independence of the mesh.

Study [18] performed a theoretical thermal analysis of a compact heat exchanger, evaluating its performance with and without water condensation on the air side. Water vapor condensation generates a film on the surface of the fins, affecting heat exchange by partially or completely covering these surfaces. The study examined various geometric configurations, considering fin surfaces and tube shapes, such as traditional round and flattened ones. The results showed the efficiency of the fins in dry (no condensation) and wet (covered by condensate) conditions, with a decrease in efficiency between 16 % and 41 %, depending on the fin geometry, validating the proposed methodology as an effective tool for analyzing and characterizing heat exchangers.

Study [19] investigated the impact of brake disc geometry on its thermal and aerodynamic performance using CFD. Four configurations were analyzed: standard full rotor (SFR), circular pillar rotor (CP), straight radial vane rotor with round vanes (SRV-R), and an innovative Y-shaped design. The results indicated that the new design improved heat dissipation, with a heat transfer coefficient (HTC) up to 10 % higher than conventional models and a 23 % increase in average air velocity. This optimization reduces material degradation and the risk of failure,

offering a significant advance in the thermal and mechanical efficiency of automotive brakes.

Despite extensive research on the use of thermal analysis and CFD simulation to improve the performance of braking systems, such as [20], there are few studies on high-power drawworks, especially 650 HP. This, in turn, means that although engineers have come closer to a solution regarding the heat dissipation factor, there are few solutions that consider all the variables to optimize the cooling of drum brakes on agricultural and industrial machinery. Currently, there are few studies that demonstrate this.

This work seeks to introduce an improvement in cooling system design for 650 HP drawworks operating braking by using CFD simulations to analyze and optimize thermal patterns in operating braking. In addition to improving the efficiency of the cooling system, this research also aims to reduce the ongoing risks of mechanical failure of the drawworks brakes, operating costs, and increase workplace safety, other areas in which the existing literature is deficient in terms of attention and detail.

II. MATERIALS AND METHODOLOGY

The objective of this article is to improve the efficiency of the water-based cooling system of the drum-type brakes of a 650 HP drawworks. For this purpose, an analysis of the current cooling system is made, which can be seen in the Fig. 3, which shows that the cooling is done by pumping water at a flow rate of 15 m³/hour in a closed circuit from a water reservoir of large dimension 6 x 2.5 x 2 m, or 30 m³. The properties used for all the simulations are water with following properties.

- $\rho = 998,2 \text{ [Kg/m}^3\text{]}$
- $C_p = 4182 \text{ [J/Kg.K]}$
- $k_p = 0,6 \text{ [W/m.Kg]}$
- $\mu = 0,001003 \text{ [Kg/m.s]}$

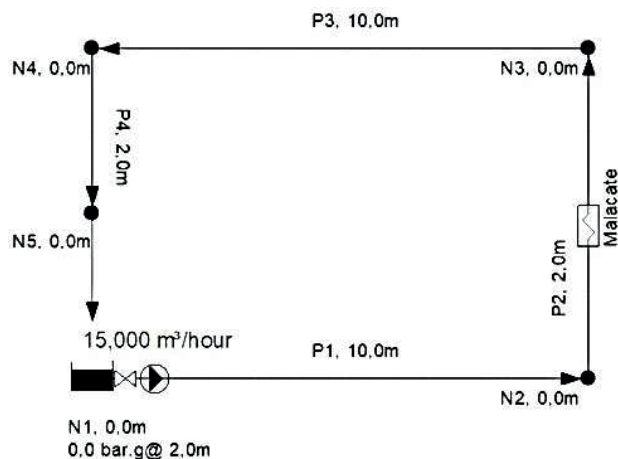


Fig. 3. Schematic of drawworks cooling system

Using the ANSYS Student software, a three-dimensional model will be developed, to subsequently perform the meshing with polygonal elements and finally run a transient state simulation to determine the temperature increase and the pressure drop of the cooling water considering the following boundary conditions in the drawworks drum:

- Water mass flow rate is 4,15 Kg/s.
- Turbulent intensity is 5 %.
- Inlet temperature is 27 °C.
- Roughness height is 0,046 mm.
- Temperature of braking Surface is 300 °C.
- Temperature of internal walls is 30 °C.
- Temperature of internal pipes is 30 °C.

Once the outlet temperature of the water in the drum of the drawworks is determined, it will be proceeded to determine the effect of thermal damping that has the 30 m³ tank has in the cooling system. For this, by means of ANSYS Student software, the three-dimensional model of the tank will be made, to subsequently perform the meshing with polygonal elements, and thus run a transient - state simulation to determine what temperature decreases after 16 hours under operation in transient state. The boundary conditions of this simulation are the following:

- Water mass flow rate is 4,15 Kg/s.
- Turbulent intensity is 5 %.
- Intel temperature is 54 °C.
- Walls temperature is 28 °C.

On the other hand, knowing the outlet temperature of the water in the drum of the drawworks, it will be proceeded to determine the amount of energy that needs to be extracted from the water to reduce the temperature to its initial condition. In this way, it will be proceeded to select a water/air crossflow heat exchanger that meets the energy and dimensional specifications of the system.

Once the heat exchanger is selected, it will be introduced into the refrigeration system as shown in the Fig. 4, and it will be proceeded to perform, by means of ANSYS Student software, the three-dimensional modeling of the heat exchanger. The meshing will be done with polygonal elements, and in this way a transient-state simulation will be run to determine that the temperature decreases in a similar way to that obtained with the 30 m³ tank. The boundary conditions for the heat exchanger are the following:

- Water mass flow rate is 4,15 Kg/s.
- Turbulent intensity is 5 %.
- Intel temperature is 54 °C.
- Roughness height is 0,046 mm.
- Heat transference rate is 100 000 W/m²

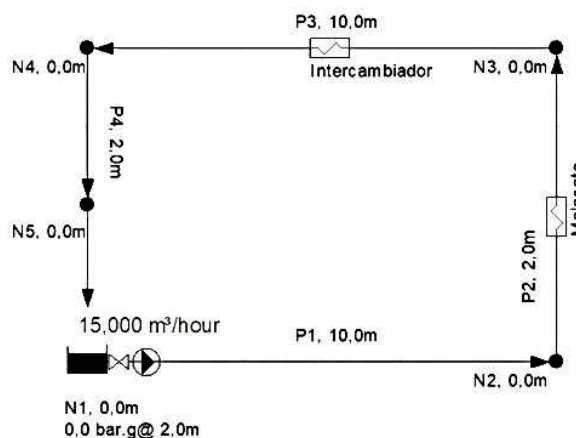


Fig. 4. Schematic of drawworks cooling system including heat exchanger

After the heat exchanger, the water will be deposited in a 6 m³ water tank, so knowing the outlet temperature of the heat exchanger, by means of ANSYS Student software, the three-dimensional model of the tank will be made, to subsequently perform the meshing with polygonal elements, and thus run a simulation of transient - state to determine what temperature decreases after 16 hours under operation in transient - state.. The boundary conditions of this simulation are:

- Water mass flow rate is 4,15 Kg/s.
- Turbulent intensity is 5 %.
- Intel temperature is 32,5 °C.
- Walls temperature is 28 °C

A. Equations and math expressions

The modeling of the entire cooling system and its elements will be done by analyzing the circulation of water as a refrigerant through the entire system of pipes, tanks and heat exchangers, so in (1) is shown the expression of continuity and in (2) the expression that is met for incompressible fluids [21]:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (1)$$

Where:

$$\nabla \cdot \vec{v} = 0 \quad (2)$$

The equation of quantity of motion is shown in the following expression (3) [22]:

$$\rho \left(\frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v} \right) = -\nabla P + \mu \nabla^2 \vec{v} + \vec{f} \quad (3)$$

Whereas the expression for the conservation of energy can be seen in (4) [23]:

$$\rho c_p \left(\frac{\partial T}{\partial t} + \vec{v} \cdot \nabla T \right) = \nabla \cdot (k \nabla T) + \dot{q} \quad (4)$$

Where:

- ρ : water density [kg/m³].
- \vec{v} : flow velocity [m/s].
- P : pressure [Pa]
- μ : dynamic viscosity [kg/m.s]
- T : temperature [K]
- c_p : specific heat at constant pressure [J/kg.K]
- k : conductivity t: thermal conductivity [W/m.K]

On the other hand, for the turbulent state, k- ω SST model will be used, since it lends the powers of the k- ω model near the walls and generates an acceptable transition to the k- ϵ model in the far region of the flow, thus giving us an advantage of versatility for an acceptable simulation of the different elements such as pipes, drawworks drum, rectangular tanks, and the heat exchanger.

The equation for turbulent kinetic energy Kappa is shown in the expression (5) [24]:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = P_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_k \mu_t \right) \frac{\partial k}{\partial x_j} \right] \quad (5)$$

The equation for specific frequency ω , is shown in the expression (6) [25]:

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\mu_j \omega)}{\partial x_j} = \alpha \frac{\omega}{\kappa} P_\kappa - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\omega \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] + 2 \left(1 - F_1 \right) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (6)$$

B. Meshing of the different elements.

By means of the data obtained, it is intended to determine the impact of introducing a heat exchanger in the cooling cycle of the drawworks drum. In this way, the present article will analyze the drawworks drum, the 30 m³ tank and the heat exchanger.

The meshing process will be configured on the polygonal - hexacore basis in order to have smoothed finite elements, reducing the pressure and temperature oscillations at cell borders [26]:

The meshing of the water in the drawworks drum can be seen in Fig. 5. Drawworks drum water mesh, which, checking the mesh quality, has a mesh quality of 0,289, with a total of 240 218 cells.

On the other hand, the mesh of the 30 m³ tank, which can be seen in Fig. 6. Mesh of 30 m³ water tank., has a quality of 0,375 with a total of 201 869 generated cells, which is a very good mesh quality.

The heat exchanger has a mesh quality of 0,347 with a total of 585 596 cells, which is also a very good mesh quality, as shown in Fig. 7. Heat exchanger mesh.

Finally in the Fig. 8. Mesh of 6 m³ water tank., the minimum orthogonal quality is 0,306 in 83 011 generated cells. All these values can be contrasted to those shown in Table I.

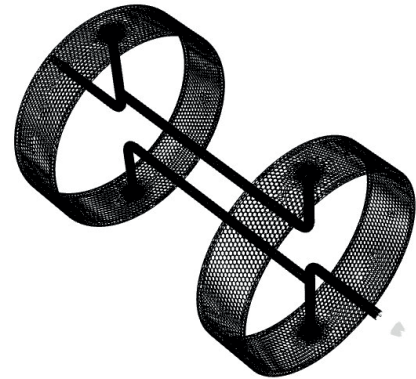


Fig. 5. Drawworks drum water mesh.

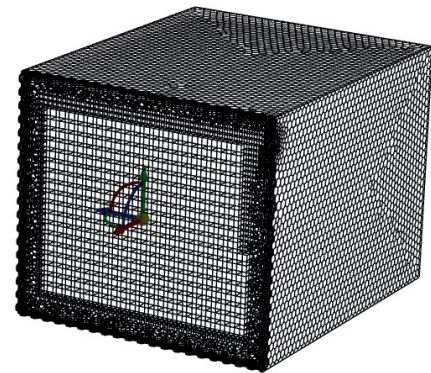


Fig. 6. Mesh of 30 m³ water tank.

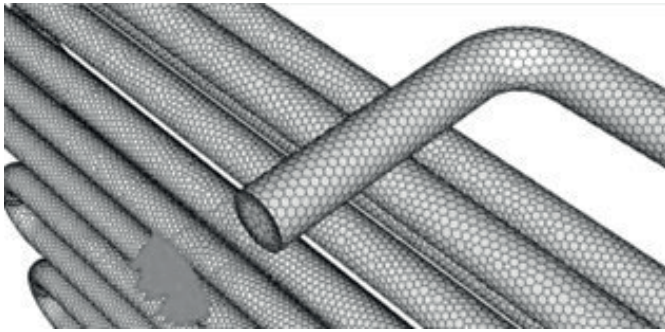


Fig. 7. Heat exchanger mesh

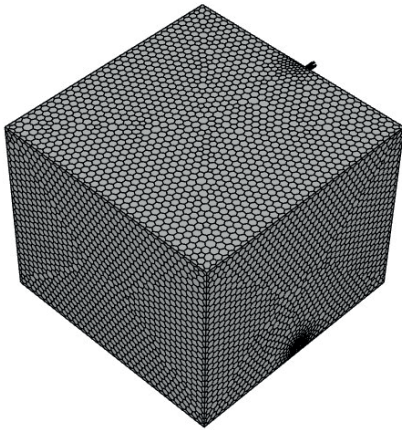


Fig. 8. Mesh of 6 m³ water tank.

TABLE I
VOLUMETRIC MESH GENERATION PARAMETERS IN ANSYS

Parameter	Element			
	Drum	Tank 30 m ³	Heat exchanger	Tank 6 m ³
Volumetric mesh creation time (min)	0,82	0,65	0,53	0,35
Cells creation time (min)	0,83	0,67	0,55	0,32
Minimum Orthogonal quality of the mesh	0,289	0,375	0,347	0,306
Total cells generated	240 218	201 869	585 596	83 011

III. RESULTS AND DISCUSSION

To analyze the thermal behavior of the brake system of the 650 hp drawworks, a computational simulation was carried out using ANSYS Student software, a tool that allows dynamic modeling of heat transfer phenomena by means of CFD. This simulation is the basis for differentiating the performance of the system in its initial configuration and evaluating the results when a heat exchanger is added. This numerical analysis helps identify the critical points of thermal accumulation and validate the need for a proposal to reduce the amount of water used to cold the drawworks braking system.

A. Simulation of water in the drawworks brake drum

For the transient-state CFD simulations, 100 iterations of 10 seconds were considered. With this information, the thermal performance of the drawworks cooling system under stationary conditions will be validated.

The Fig. 9 shows that initially, a temperature close to 135 °C is observed, which decreases quickly as the system enters the regime. In less than 100 seconds, the thermal gradient begins to stabilize, reaching a temperature of approximately 57,5 °C at the end of the simulation. This decreasing indicator affirms a successful heat dissipation process, in which the cooling fluid fulfills the function of absorbing the thermal energy presented in the drum.

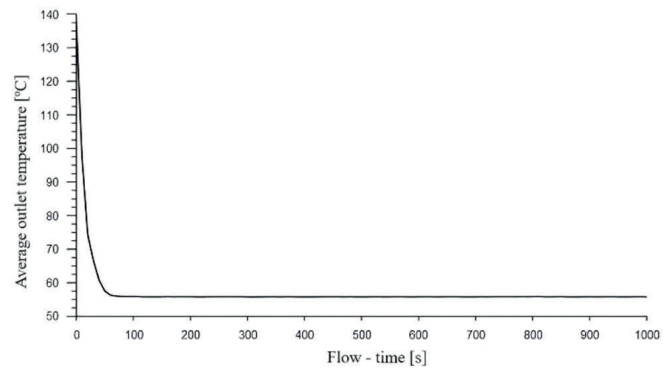


Fig. 9. Average water temperature at the outlet of the drum.

Fig. 10 shows the evolution of the pipe heat variable, which represents the amount of heat exchanged between the pipe and walls of the cooling system and the fluid. An initial zero value is observed, and it increases until it stabilizes at 120 000 W/m². This result represents the heat flow out of the pipe and walls to the cooling fluid, which is admitted into the cooling system. The positive value means a net heat flow dissipated from the metal structure of the drum to the fluid, with the progressive stabilization of the value.

In the CFD analysis, the fluid trajectories from the inlet to the outlet of the cooling system are represented. The color scale goes from 27 °C to more than 100 °C as shown in Fig. 11. The fluid enters at low temperature (blue color) and heats up as it circulates inside the drums (green, yellow, and red colors).

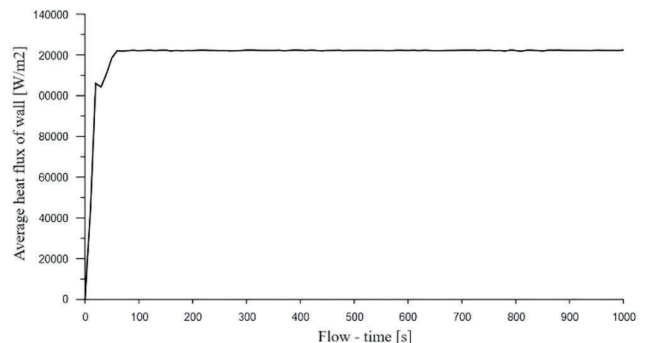


Fig. 10. Average heat flux in the water circulating by drum.

The green color indicates that the fluid in the zone reaches a temperature between 54 °C and 56 °C. This confirms that at the end of the system, the refrigerant absorbs a significant amount of heat from the drums. The correct color distribution also indicates that the thermal mixing is correct, with no thermal stratification in the outlet section, as shown in Fig. 12.

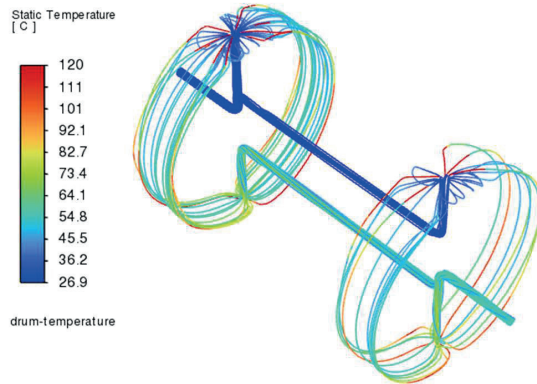


Fig. 11. CFD analysis of water circulation by drum.

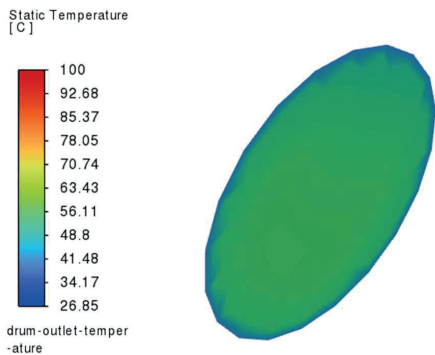


Fig. 12. Temperature surface at drum outlet

The velocity inside the drawworks drum varies between near zero inside the drawworks drum and rises to 5,4 m/s in the curvatures of the pipe. On the other hand, in the inlet and the outlet of the drum, the velocity is more stable with a 4,15 m/s, which corresponds to the mass conservation equation. This is shown in Fig. 13.

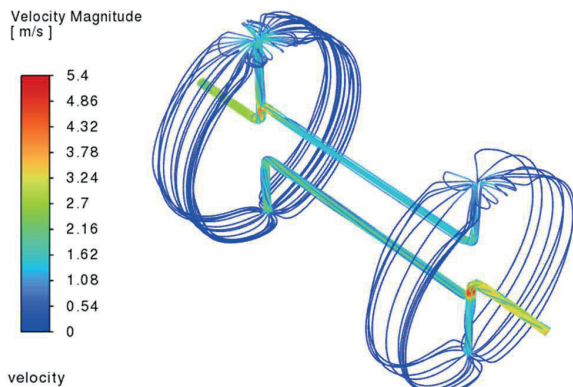


Fig. 13. CFD water velocity inside the drum.

Fig. 14 shows that during the 1 000 seconds of flow time the pressure drop value remains practically constant, stabilizing at -13 317 Pa, which indicates that the modeling has reached stationary conditions, without variations that affect the pressure in the outlet zone. Therefore, it has achieved hydraulic equilibrium under the boundary conditions, which confirms that there are no cavitation phenomena or pressure accumulation in any sector that could negatively affect the modeling

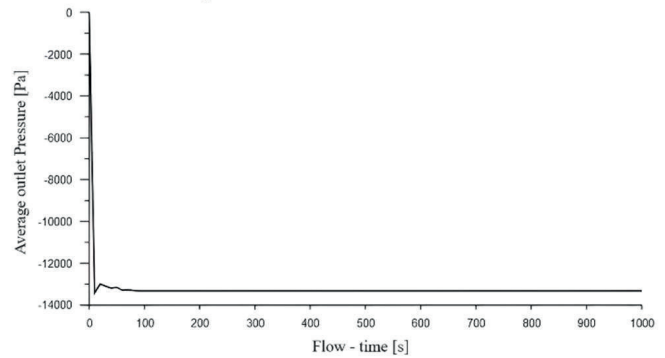


Fig. 14. Pressure drop in brake drum.

B. Simulation of water in the 30 m³ tank

This tank will act as a thermal reservoir and pressure regulator; it will maintain a continuous supply to the cooling system during braking cycles. The meshing will allow simulating phenomena such as thermal stratification, internal recirculation, and temperature distribution. In the case of the tank, since it handles a large amount of fluid, a transient flow analysis will be carried out, since having a 15m³/h pump, when the water enters the tank with this flow rate, a dynamic and thermal damping is formed.

Fig. 15 shows the behavior of the fluid outlet temperature during a transient CFD simulation of 60 000 seconds (16 hours). It is visualized that, after a small initial rise, the average outlet temperature stabilizes when it reaches 31,5 °C, with small oscillations that are maintained in the range considered. This indicates that the system reaches a constant transient thermal regime.

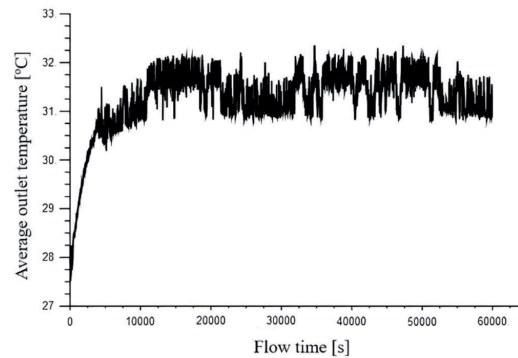


Fig. 15. 30 m³ tank outlet water temperature.

The transient analysis confirms that this state could only be reached after a prolonged operation zone. Therefore, Fig. 16,

where the heat flux is almost stable and reaches the value of -4500 W/m^2 . The equilibrium thermal regime is not achieved in the determined time, which is 16 hours, which indicates that, during the normal operation cycles of the drawworks, the outlet temperature remains stable and does not present relevant increases that compromise the thermal performance of the system.

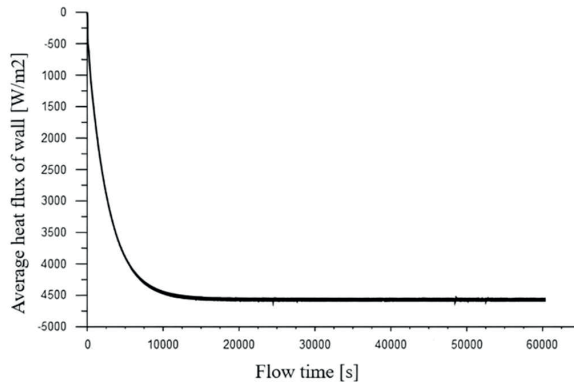


Fig. 16. Thermal regime heat-walls of 30 m^3 tank.

C. Simulation of heat exchanger

This paper proposes the incorporation of a heat exchanger in the circuit, which will allow maintaining low and controlled temperatures without depending exclusively on the volume of water in the tank. The purpose of the heat exchanger is to efficiently transfer heat from the hot fluid to an external cooling medium, achieving a more constant heat dissipation in less time.

Fig. 17 shows the evolution of the area-weighted average temperature at the heat exchanger outlet. The curve shows a rapid decrease from the initial $54 \text{ }^\circ\text{C}$ to stabilize around $32,5 \text{ }^\circ\text{C}$ after approximately 100 seconds, indicating that the system reaches the transient state efficiently, ensuring effective heat transfer to the fluid using a $400\,000 \text{ W}$ heat exchanger.

It is visualized how the fluid temperature changes as it moves through the heat exchanger. The inlet temperature is close to $54 \text{ }^\circ\text{C}$ (marked in red) and the outlet temperature is approximately $32 \text{ }^\circ\text{C}$ (in green), as shown in Fig. 18. This temperature change demonstrates that there is effective heat transfer to the surroundings, which supports the good performance of the design under transient - state conditions.

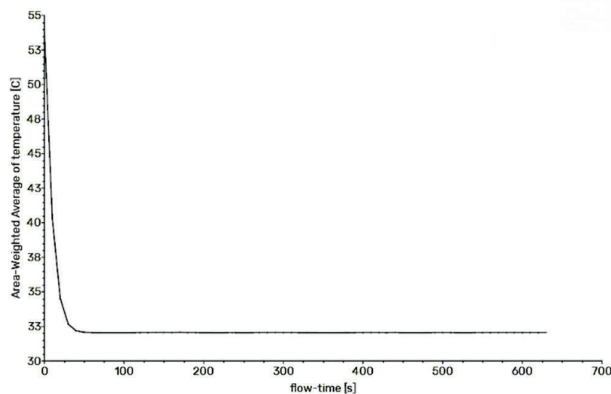


Fig. 17. Stabilization of average temperature at the heat exchanger outlet.

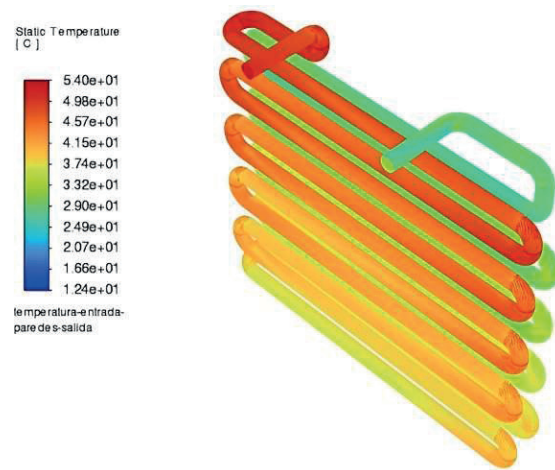


Fig. 18. Heat exchanger circulating water temperature lines.

The weighted average of the pressure value at the heat exchanger outlet, stabilizes around $-43\,000 \text{ Pa}$ after the first 200 seconds, as it is shown in Fig. 19. This trend indicates the transient flow regime, where pressure variations are significantly reduced, ensuring stable outlet conditions for the thermal analysis of the system.

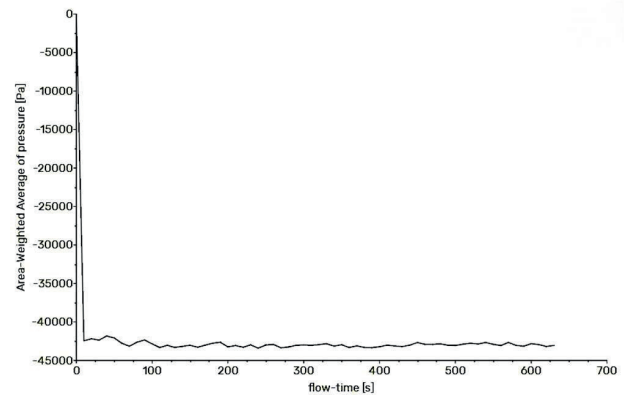


Fig. 19. Pressure drop in heat exchanger.

D. Simulation of water in the 6 m^3 tank

This tank will also act as a thermal reservoir and pressure regulator; it will maintain a continuous water supply to the cooling system during braking cycles. The meshing will allow simulating phenomena such as thermal stratification, internal recirculation, and temperature distribution. A transient flow analysis will be carried out, since having a $15 \text{ m}^3/\text{h}$ pump, when the water enters the tank with this flow rate, a dynamic and thermal damping is formed.

The behavior of the fluid outlet temperature during a transient CFD simulation of 60 000 seconds (16 hours) is shown Fig. 20 is visualized that, after a small initial rise, the average outlet temperature stabilizes when it reaches $30 \text{ }^\circ\text{C}$, after about 100 seconds, with small oscillations that are maintained in the range considered. This indicates that the system reaches a constant transient thermal regime.

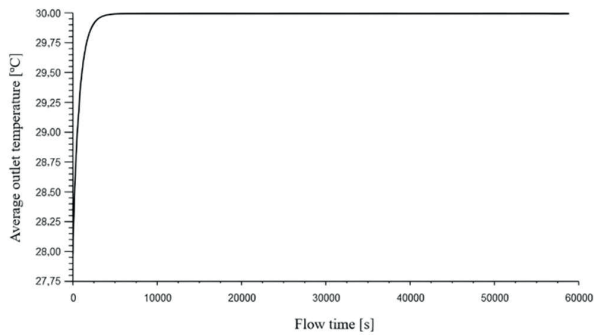


Fig. 20. 6 m³ tank outlet water temperature.

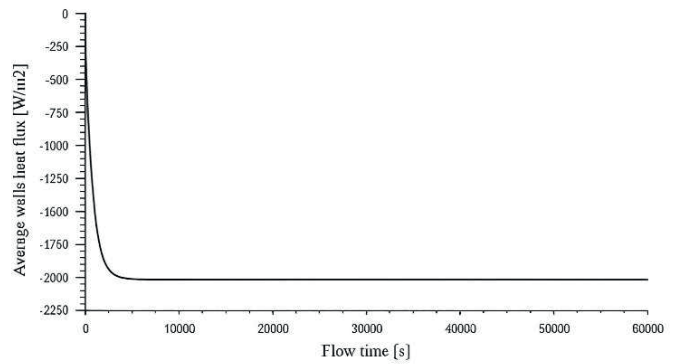


Fig. 21. Thermal regime heat-walls of 6 m³ tank.

The transient analysis confirms that this state could only be reached after a prolonged operation time like, where the heat flux is almost stable, as shown in Fig. 21. The equilibrium thermal regime is not achieved in the determined time, which is 16 hours, which indicates that, during the normal operation cycles of the drawworks, the outlet temperature remains stable and does not present relevant increases that compromise the thermal performance of the system.

E. PipeFlow simulation without and with heat exchanger.

The cooling system has a 2,2 kW centrifugal pump, with an outlet pressure of 50 m for a flow rate of 15 m³/h, which is sufficient to overcome the pressure drop of all the piping and drawworks until the circuit is closed in the reservoir tank. As shown in Fig. 22 the final pressure of the system is 3,53 bar, with a constant water velocity of 3,17 m/s. When the heat exchanger is added to the system the final pressure is 3,18 bar, with the same 3,17 m/s water velocity. This can be corroborate in the Fig. 23.

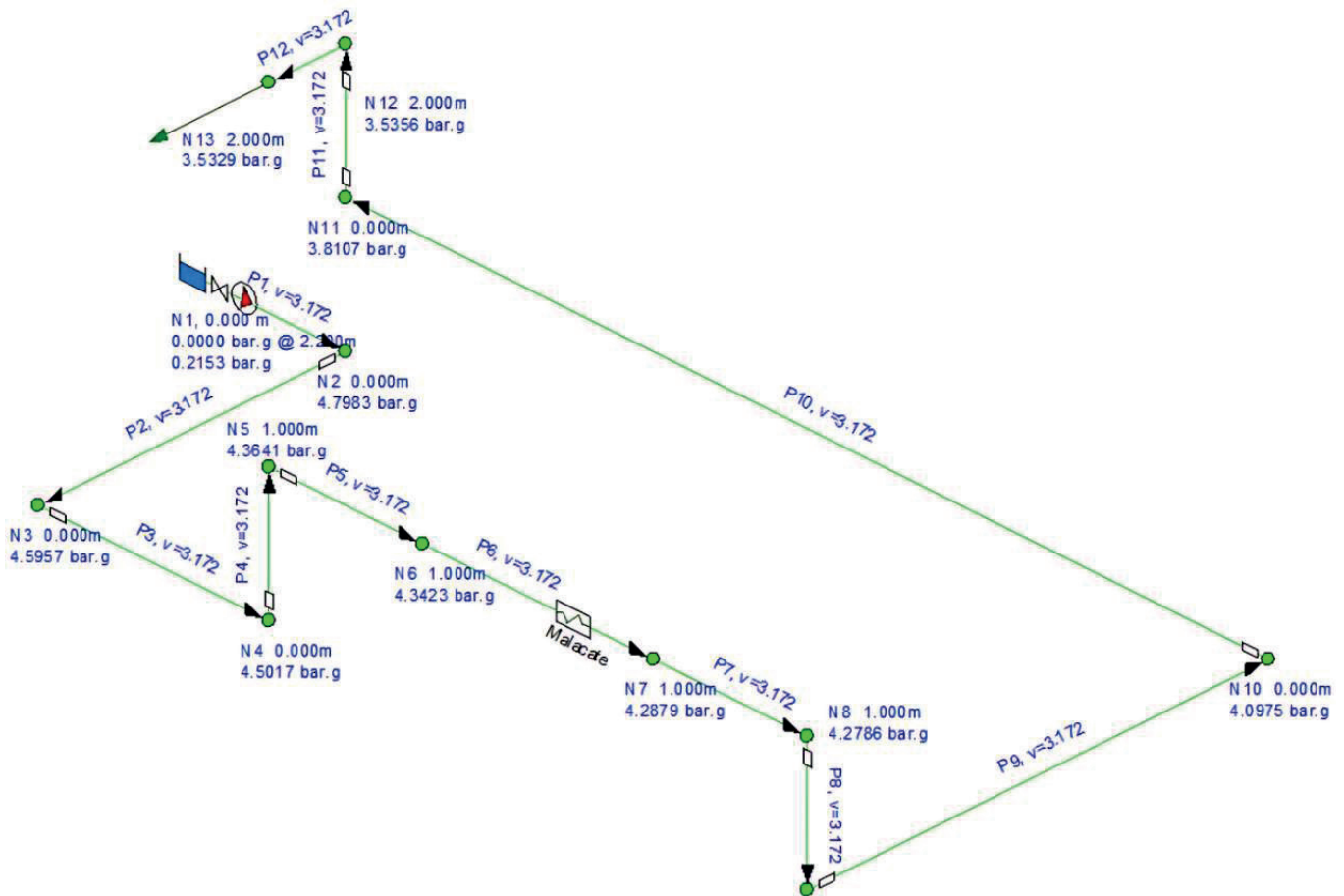


Fig. 22. Line flow diagram for drawworks drum cooling system.

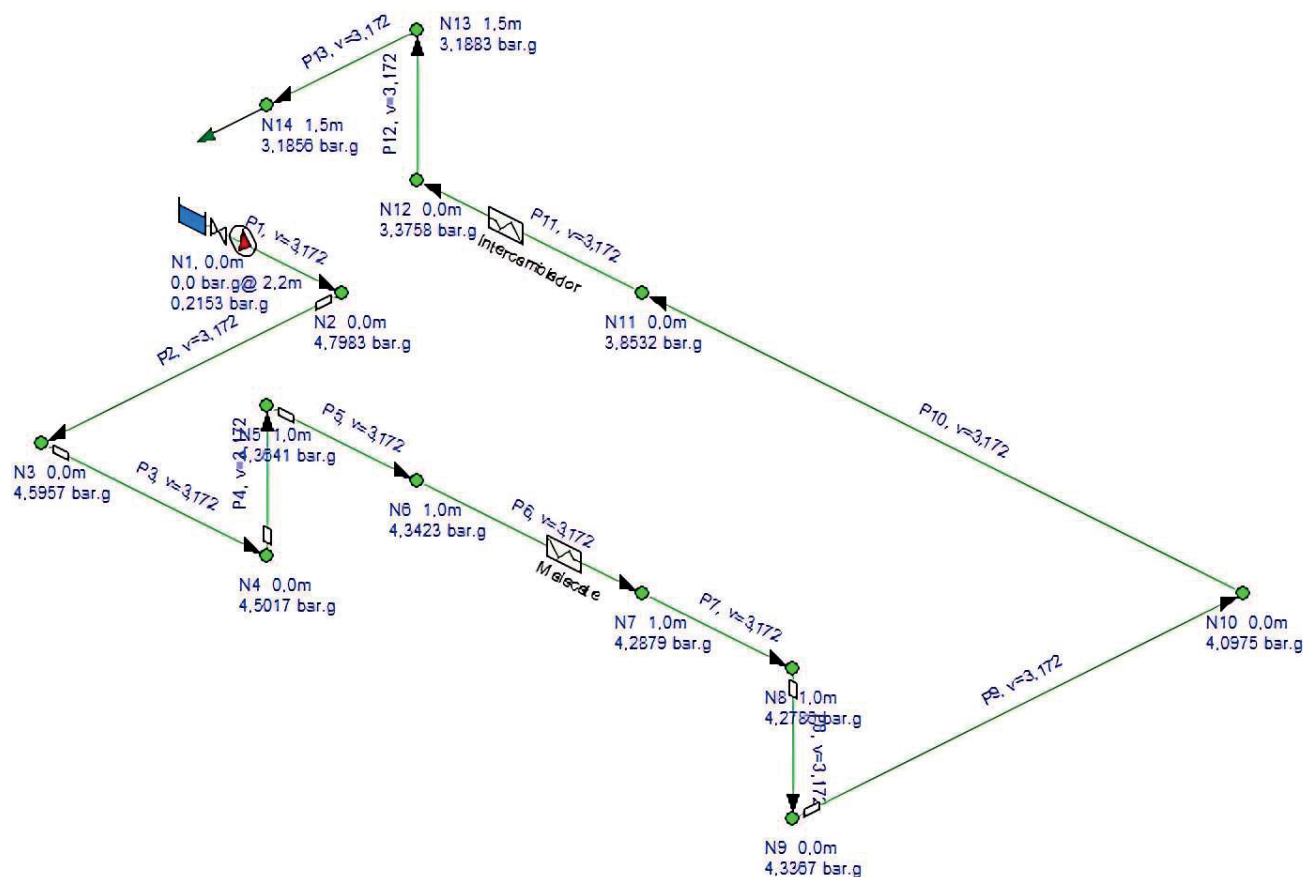


Fig. 23. Line flow diagram for drawworks drum cooling system with heat exchanger added.

IV. CONCLUSIONS

This study has demonstrated that the implementation of a cooling system, based on a cross-flow water-air heat exchanger with a capacity of 400 kW, represents a technically feasible and thermally efficient solution for temperature control in 650 HP drawworks brake drums used in oil refurbishment operations.

The main conclusions are summarized below:

- **Increased thermal efficiency:** the fluid temperature in the improved system was reduced to 32 °C, compared to 56 °C achieved with the traditional system, representing a significant improvement in heat dissipation.
- **Optimization of storage volume:** the use of a 6 m³ tank instead of the traditional 30 m³ tank reduces the volume of water required by 80 %, without compromising thermal or hydraulic performance.
- **Reduced thermal stabilization time:** the improved configuration allows stable conditions to be reached in less operating time, optimizing work cycles and increasing the reliability of the braking system.
- **Successful hydraulic validation:** the simulation with Pipe Flow Expert confirmed that a 2,2 kW centrifugal pump can sustain the proposed system, with an effective pressure of 3,18 bar, guaranteeing its operability without cavitation or overloading risks.
- **Contribution to sustainable design:** by considerably reducing water consumption and the volumes required

for cooling, the proposed system constitutes a more compact, efficient and environmentally friendly alternative, aligned with sustainable engineering practices.

- **Future designs:** it should be noted that the use of appropriately sized heat exchangers in cooling systems helps to increase their thermal efficiency. In addition, it would be advisable to conduct research using different refrigerants that could help to increase the efficiency of this cooling system and allow its use in different types of machinery.

Based on these results, it is concluded that the integration of a heat exchanger in high-power industrial brake cooling systems is not only technically and economically feasible, but also offers tangible benefits in terms of energy efficiency, resource reduction, and operational stability. Future research can address the dynamic behavior in variable-load scenarios and consider the implementation of intelligent control systems to regulate heat transfer in real time.

REFERENCES

- [1] C. Li, D. Wang, Y. Sun, W. Xu, J. Zhang, K. Wu, J. Chen, H. Deng, D. Zhang, and S. Ge., "Bending fatigue damage behavior of wire rope in hoisting system of drilling rig," *Tribol. Int.*, vol. 187, p. 108745, 2023, doi: <https://doi.org/10.1016/j.triboint.2023.108745>.
- [2] D. Pavković, M. Cipek, M. Krznar, and J. Benić, "A retrofitting control system design suitable for deep borehole drilling using legacy drawworks mechanical brake hardware," *Energy Convers. Manag.*, vol. 260, p. 115589, 2022, doi: <https://doi.org/10.1016/j.enconman.2022.115589>.

- [3] A. Labdi, and M. Bouchetara, "Optimization of the Thermal and Mechanical Behavior of an Aeronautical Brake Disc using the Design of Experiments Approach," *MECHANIKA*, vol. 24, no. 2, pp. 191-196, 2018, doi: <https://doi.org/10.5755/j01.mech.24.2.19110>
- [4] F. Yusubov, and A. Aliyev, "Temperature distribution in a drilling brake contact," *J. Tribol.*, vol. 31, pp. 28-45, 2021.
- [5] J. J. Azar, "Oil and Natural Gas Drilling," in *Encyclopedia of Energy*, C. J. Cleveland, Ed., New York: Elsevier, 2004, pp. 521-534. doi: <https://doi.org/10.1016/B0-12-176480-X/00254-0>.
- [6] M. Kindrachuk, D. Volchenko, N. Fidrovskaya, O. Dukhota, D. Zhuravlev, M. Ostashuk, Y. Porokhovskiy, and V. Kharchenko, "Wear- Friction Properties Of Friction Pairs in disc-pad brakes," *East.-Eur. J. Enterp. Technol.*, vol. 4, no. 12(124), pp. 56-61, 2023, doi: <https://doi.org/10.15587/1729-4061.2023.285699>.
- [7] J. I. Rojas-Sola and J. Molino-Delgado, "3D Parametric design and static analysis of the first Spanish winch used to drain water from mines: [Diseño paramétrico tridimensional y análisis estático del primer malacate español utilizado para drenar agua de las minas]," *DYNA (Colombia)*, vol. 82, no. 190, pp. 237-245, 2015. doi: <https://doi.org/10.1007/s12206-023-0342-8>
- [8] Y. Cai, W. Tang, Q. Huang, and L. Wang, "Finite element analysis (FEM) on thermal fatigue failure of water-cooling disc brake," *Zhongguo Jixie Gongcheng/China Mech. Eng.*, vol. 23, no. 22, pp. 2726-2731, 2012, doi: <https://doi.org/10.3969/j.issn.1004-132X.2012.22.016>.
- [9] R. K. Singh and C. Sarkar, "In-situ braking torque and temperature analysis of two-wheeler drum brake in friction braking," *J. Mech. Sci. Technol.*, vol. 37, no. 4, pp. 2069-2077, 2023, doi: <https://doi.org/10.1007/s12206-023-0342-8>.
- [10] S. Zhang, J. Wang, Z. Mu, Y. Zhang, and P. Yang, "Research on water cooling control strategies and factors influencing disc-type eddy current brakes," *Case Stud. Therm. Eng.*, vol. 75, p. 107089, 2025, doi: <https://doi.org/10.1016/j.csite.2025.107089>.
- [11] J. Zhang, G. Hu, Q. Cheng, L. Yu, and W. Zhu, "Analysis of braking performance and heat dissipation characteristics of multi-disc magnetorheological brake with an inner water-cooling mechanism," *J. Magn. Magn. Mater.*, vol. 604, p. 172313, 2024, doi: <https://doi.org/10.1016/j.jmmm.2024.172313>.
- [12] A. A. De la Rosa-Andino, I. Macías Socarrás, Y. Morales Tamayo, D. Pérez Sutil, I. Rodríguez Beltrán, and J. A. Montaguano Toaquiza, "Proposal for Redesign of a Winch for Pulling Loads," *Rev. Cienc. Téc. Agropecu.*, vol. 29, pp. 84-96, Sept. 2020.
- [13] A. Begag, R. Saim, S. Abboudi, and H. F. Öztıp, "Effect of internal and external corrugated surfaces on the characteristics of heat transfer and pressure drop in a concentric tube heat exchanger," *Int. J. Therm. Sci.*, vol. 165, p. 106930, 2021, doi: <https://doi.org/10.1016/j.ijthermalsci.2021.106930>.
- [14] M. A. S. S. Ravnani, R. Cardoso, and A. P. Da Silva, "A new heat exchanger network for a soybean oil extraction plant using pinch analysis," *Inf. Technol.*, vol. 12, no. 2, pp. 81-86, 2001.
- [15] R. Chucuya, L. Narvaez, W. Marino Vila, D. Rodríguez, and L. F. Saldaña Bernuy, "Numerical evaluation of heat transfer parameters in shell and tube exchangers using water-based TiO₂ nanoparticles," in *Proceedings of the LACCEI international Multi-conference for Engineering, Education and Technology*, César Vallejo University, UCV Campus, Chimbote, Peru: Latin American and Caribbean Consortium of Engineering Institutions, 2024. doi: <https://doi.org/10.18687/LACCEI2024.1.1.1891>
- [16] F. Toapanta, J. Cortez, W. Quitiaquez, and W. Orellana, "Estudio Numérico Mediante CFD del Proceso de Enfriamiento con Intercambiadores de Calor en Sistemas Computacionales," *Rev. Téc. Energ.*, vol. 17, pp. 55-64, 2021.
- [17] L. Sáez De Argandoña Lazpita, "Estudio de independencia del malla-do en simulación de la recirculación del flujo por CFD." Universidad del País Vasco, Nov. 30, 2023. [Online]. Available: <http://hdl.handle.net/10810/63283>
- [18] M. T. Martínez, E. Torres, and J. A. Soto, "Evaluación de Intercambiadores de Calor Compactos de Tubos Aletados TT - Evaluation of Compact Finned-Tube Heat Exchangers," *Inf. Tecnológica*, vol. 15, no. 4, pp. 47-54, 2004.
- [19] M. B. J. Haddar, A. Ghorbel, F. Djemal, M. Baccar, and M. Haddar, "Heat transfer and flow analysis through a new brake disc design: a CFD approach," *Int. J. Interact. Des. Manuf. IJIDeM*, 2024, doi: <https://doi.org/10.1007/s12008-024-02027-2>
- [20] A. Labdi and M. Bouchetara, "Optimization of the Thermal and Mechanical Behavior of an Aeronautical Brake Disc using the Design of Experiments Approach," *MECHANIKA*, vol. 24, no. 2, pp. 191-196, 2018, doi: <https://doi.org/10.5755/j01.mech.24.2.19110>.
- [21] H. İ. Variyenli, M. Yousefi, A. D. Tuncer, A. Sözen, B. Ç. Göksel, and A. Khanlari, "Numerical and experimental analysis of a water cooled photovoltaic-thermal system using modified tubes," *Appl. Therm. Eng.*, p. 127559, 2025, doi: <https://doi.org/10.1016/j.applthermaleng.2025.127559>.
- [22] W. Wang, L. Chen, Y. Kong, L. Yang, Y. Niu, and X. Du, "Cooling performance evaluation for double-layer configuration of air-cooled heat exchanger," *Int. J. Heat Mass Transf.*, vol. 151, p. 119396, 2020, doi: <https://doi.org/10.1016/j.ijheatmasstransfer.2020.119396>.
- [23] M. Khaled, "Enhancing thermal performance of water-air cross flow heat exchangers through upstream nozzle design and unit division," *Int. J. Thermofluids*, vol. 27, p. 101223, 2025, doi: <https://doi.org/10.1016/j.ijft.2025.101223>.
- [24] F. Toapanta-Ramos, J. Espin, and W. Diaz, "Numerical Study Using CFD on Heat Sinks for Electronic Components," *Enfoque UTE*, vol. 16, pp. 41-48, June 2025.
- [25] J. Wu, X. Su, Z. Gong, J. Cai, and L. Gu, "Exploration of high-efficiency heat transfer modes in lead-cooled assemblies: A structural optimization based on the SST $k-\omega-k\theta-\epsilon\theta$ model," *Appl. Therm. Eng.*, vol. 247, p. 123051, 2024, doi: <https://doi.org/10.1016/j.applthermaleng.2024.123051>.
- [26] M. Liu, G. Gao, H. Zhu, C. Jiang, and G. Liu, "A cell-based smoothed finite element method (CS-FEM) for three-dimensional incompressible laminar flows using mixed wedge-hexahedral element," *Eng. Anal. Bound. Elem.*, vol. 133, pp. 269-285, 2021, doi: <https://doi.org/10.1016/j.enganabound.2021.09.008>